

“Numerical Analysis in Ranque-Hilsch Vortex Tube.”

A THESIS SUBMITTED IN PARTIAL FULFILLMENT
OF THE REQUIREMENTS FOR THE DEGREE OF

**Bachelor of Technology
In
Mechanical Engineering**

**By
Shreetam Dash
10603028**

Under the Guidance of
Prof. R. K. Sahoo



**Department of Mechanical Engineering
National Institute of Technology
Rourkela
2010**



Acknowledgment

We deem it a privilege to have been the student of Mechanical Engineering stream in National Institute of Technology, ROURKELA.

Our heartfelt thanks to Dr.R.K.Sahoo, our project guide who helped us to bring out this project in good manner with his precious suggestion and rich experience.

I would like to render heartiest thanks to Md. Ezaz Ahammed who's ever helping nature and suggestion has helped me to complete this present work.

We take this opportunity to express our sincere thanks to our project guide for cooperation in accomplishing this project a satisfactory conclusion.

Submitted by:

Shreetam Dash

Roll. 10603028

Department of Mechanical Engg

National Institute of technology



**National Institute of Technology
Rourkela
CERTIFICATE**

This is to certify that the thesis entitled “**Numerical Analysis in Ranque-Hilsch Vortex Tube**” submitted by **Shreetam Dash, Roll No: 10603028** in the partial fulfillment of the requirement for the degree of **Bachelor of Technology in Mechanical Engineering**, National Institute of Technology, Rourkela, is being carried out under my supervision.

To the best of my knowledge, the matter embodied in the thesis has not been submitted to any other university/institute for the award of any degree or diploma.

Date:

Professor R.K.Sahoo

Department of Mechanical Engineering

National Institute of Technology

Rourkela-769008

CONTENTS

Contents-----	4
List of figures-----	5
Abstract-----	6

	Page No.
1. Introduction	
1.1 Introduction-----	8
1.2 Important Definitions -----	10
1.3 Types of Vortex tube-----	11
2. Computational Fluid Dynamics	
2.1 A Working Definition of Computational Fluid Dynamics -----	14
2.2 The CFD Process -----	15
2.3 PROBLEM SOLVING STEPS-----	20
3. CFD Analysis	
3.1 CFD model-----	22
3.2 Procedure for Modeling-----	23
3.3 Analysis in Fluent-----	26
4. Results and Discussions-----	33
5. Conclusion-----	43
6. References-----	45

LIST OF FIGURES

SL NO.	TOPICS	Page NO.
1	Ranque-Hilsch counter flow vortex tube	12
2	Ranque-Hilsch uniflow vortex tube	12
3	2D CFD model of vortex tube	22
4	Grid display showing hot outlet, cold outlet and pressure inlet	34
5	Graph showing iteration and convergence	35
6	Graphs showing the Velocity vectors	36
7	Contour showing the velocity variation along inlet, cold outlet and hot outlet.	37
8	Contour showing the temperature variation along inlet, cold outlet and hot outlet	38
9	Graph showing the Temperature distribution along the vortex axis, inlet and both the outlets	39
10	Graph showing the Pressure distribution along the vortex axis, inlet and both the outlets.	39
11	Graph showing the Velocity distribution along the vortex axis, inlet and both the outlets.	40

ABSTRACT

The vortex tube, also known as the Ranque-Hilsch vortex tube, is a mechanical device that separates a gas into hot and cold streams. It has no moving parts.

The gas is injected tangentially into a chamber with high degree of swirl. At the other end because of the nozzle only the outer periphery part is allowed to escape where as the inner part is forced into the inner vortex with a small diameter within the outer vortex.

A two dimensional computational model was used to determine the energy separation and flow phenomenon. An axisymmetric model for a counter flow vortex tube was used. The model on simulation explains the temperature separation phenomena within the vortex tube. The fluid in the vortex tube has three regions. One is the fluid entering at a particular temperature and pressure which leads into two outlets, the cold exit and the hot exit where the temperature is less or more from the inlet temperature respectively. The swirl flow results in the temperature drop at the cold exit. The temperature separation occurs because of the heat transfer from the central flow to the peripheral flow. The simulation is done with fluent using the k- ϵ model.

FLUENT is software used to simulate fluid flow problems. FLUENT is generally used for computational Fluid Dynamics problems. It uses the finite-volume method to solve the governing equations for a fluid. It provides a wide field to solve problems. Geometry and grid generation is done using GAMBIT.

Chapter 1

INTRODUCTION

INTRODUCTION:

The vortex tube is a device which separates a high pressure flow entering tangentially into two low pressure flows, thereby producing a temperature change. The vortex tube has no moving parts and generally consists of a circular tube with nozzles and a throttle valve. High pressure gas enters the vortex tube tangentially through the nozzles which increases the angular velocity and thus produces a swirl effect. There are two exits in the vortex tube. The hot exit is located in the outer radius near the far end of the nozzle and the cold exit is in the centre of the tube near the nozzle. The gas separates into two layers. The gas closer to the axis has a low temperature and comes out through the cold exit and the gas near the periphery of the tube has a high temperature which comes out through the hot exit. The difference in the temperature produced due to the swirl flow was first observed by Ranque during the study of dust separation cyclone and he referred it as “temperature separation”. Other investigator has proposed the idea that the energy transfer is due to compression and expansion during the flow. The proposed mechanism includes turbulent eddies, embedded secondary circulation and vortices. Gutsol proposed that the energy separation is due to the centrifugal separation of micro volumes with different azimuthal velocity in the vortex tube.

The vortex tube was first observed by Ranque. Later, Hilsch did some experimental and theoretical studies to increase the efficiency of the vortex tube. Fulton explained the energy separation. He proposed that the inner layer heats the outer layer meanwhile expanding and growing cold. Reynold did the numerical analysis of vortex tube. Lewellen arrived at the solution by combining the three Navier-stokes equations for an incompressible fluid in a strong rotating axisymmetric flow with a radial sink flow. Linderstorm-lang examined the velocity and thermal fields in the tube. He calculated the axial and radial gradients of the tangential velocity profile

from prescribed secondary flow functions on the basis of a zero-order approximation to the momentum equations developed by Lewellen for an incompressible flow.

Schlenz investigated numerically in a uni-flow vortex tube, the flow field and the process of energy separation. Calculations were carried out assuming a 2D axisymmetric compressible flow and using the Galerkin's approach with a zero-equation turbulence model to solve the mass, momentum, and energy conservation equations to calculate the flow and thermal fields. Amitani used the mass, momentum and energy conservation equation in a 2D model of a counter flow vortex tube with a short length, with an assumption of a helical motion in an axial direction for an inviscid compressible perfect fluid. Flohlingsdorf and Unger used the CFX system with the $k-\epsilon$ model to study the velocity profile and energy separation in a vortex tube. Aljuwahel reported that RNG $k-\epsilon$ is better than $k-\epsilon$, when he studied the energy separation and the flow phenomenon in a counter flow vortex tube using CFD in Fluent. T. Farouk and B. Farouk introduced the large eddy simulation (LES) technique to predict the flow fields and the associated temperature separation within a counter-flow vortex tube for several cold mass fractions. However, most of the computations found in the literature used simple or the first-order turbulence models which are considered unsuitable for complex, compressible vortex tube flows. The present work presents a two-dimensional numerical investigation of flow and temperature separation behaviors inside a flow vortex tube.

Important definition:

A few important terms related to vortex tube has been discussed.[4]

1. Cold mass fraction:

The cold mass fraction defines the ratio between the mass of air moving out through the cold exit to the actual mass of air entering the vortex tube through the inlet. It is an important parameter which defines the performance and the temperature separation of a vortex tube. This is expressed by

$$\mu_c = M_c / M_i,$$

Where M_c is the mass flow rate of cold air and M_i is the mass flow rate through the inlet.

2. Cold air temperature drop

Cold air temperature drop is the difference between the temperatures of air at inlet to that of the temperature of air at cold exit. It is denoted by

$$\delta T_c = T_i - T_c;$$

Where T_i is the temperature of air at inlet and T_c is the temperature of air at cold outlet.

3. Cold orifice diameter ratio

Cold orifice diameter ratio is the ratio between the diameters (d) at the cold exit to that at the inlet (D).

$$\beta = d/D$$

4. Coefficient of performance

It is defined as defined as a ratio of cooling rate to the energy used in cooling. It is given by

$$\text{COP} = Q_c / W$$

Where Q_c is the cooling rate per unit of air in the inlet vortex tube, and W is mechanical energy used in cooling per unit of air inlet.

Types of Vortex tube

The vortex tube is generally classified into two types. They are uniflow type or parallel type and counter flow type.[4]

Uniflow type

This type of vortex tube has both the cold and hot exit at the far end of nozzle in the same side. In this type the vortex tube consist of a nozzle, vortex tube and the cold exit present concentrically with the annular hot exit. The main applications of the vortex tube are in those areas where compactness, reliability, and low equipment costs are the major factors and the operating efficiency is of no consequence. Some typical applications are cooling devices for airplanes, space suits and mines; instrument cooling; and industrial process coolers.

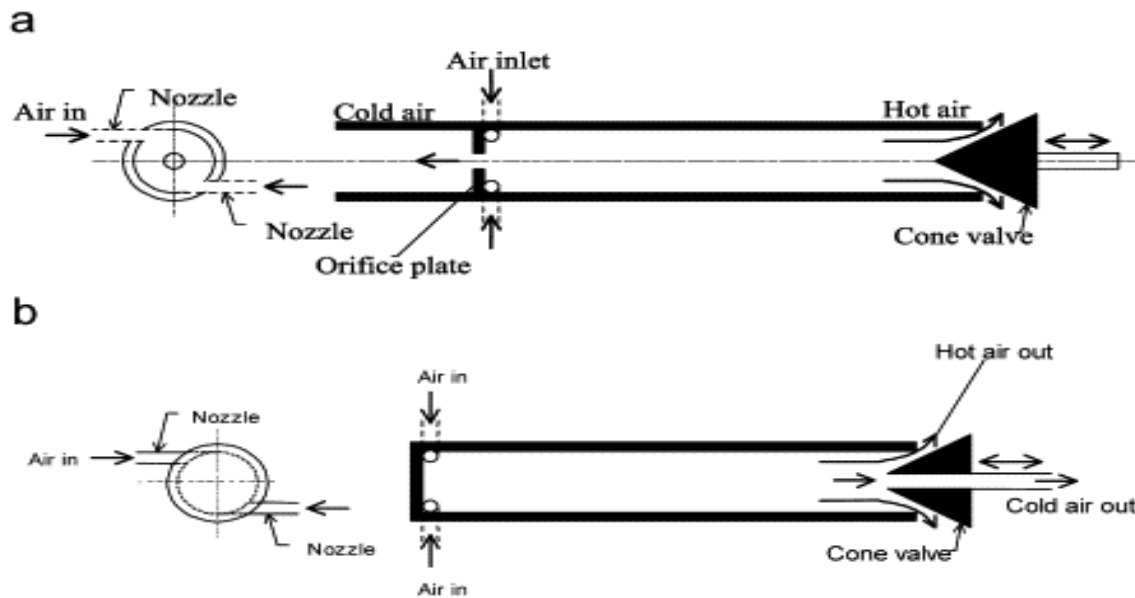


Fig 1(a) Ranque-Hilsch counter flow vortex tube. Fig1(b) Ranque-Hilsch uniflow vortex tube

Counter-flow vortex tube:

The counter flow vortex tube consists of a nozzle, vortex tube, and hot outlet with a cone shaped valve which controls the output. The cold exit is present centrally near the nozzle end. A source of compressed gas (e.g. air) at high pressure enters the vortex tube tangentially through one or more inlet nozzles at a high velocity. The expanding air inside the tube then creates a rapidly spinning vortex. The length of the tube is typically between 30 and 50 tube diameters. As the air expands down the tube, the pressure drops sharply to a value slightly above atmospheric pressure,. Centrifugal action will keep this constrained vortex close to the inner surface of the tube. The air that escapes at the other end of the tube can be varied by a flow-control valve, usually shaped as a cone. The amount of air released is between 30% and 70% of the total airflow in the tube. The remainder of the air is returned through the centre of the tube, along its axis as a counter-flowing stream. Once a vortex is set up in the tube, the air near the axis cools down while the air at periphery heats up in comparison with the inlet temperature. This phenomenon is known as temperature separation effect.

Chapter 2

Computational Fluid Dynamics

Definition of Computational Fluid Dynamics

- Computational – mathematical analysis, computing
- Fluid Dynamics - the dynamics of things that flow

CFD - a computational technology that enables one to study the dynamics of things that flow. CFD is concerned with numerical solution of differential equations governing transport of mass, momentum and energy in moving fluid. Using CFD, one can build a computational model on which physics can be applied for getting the results. The CFD software gives one the power to model things, mesh them, give proper boundary conditions and simulate them with real world condition to obtain results. Using CFD a model can be developed which can breed to give results such that the model could be developed into an object which could be of some use in our life.

The Benefits of CFD

There are three reasons to use CFD software: insight, foresight, and efficiency.

Insight

Think of an object which is difficult to produce practically and do some experiment on it. CFD provides the break through. Using the CFD software, one can easily design the object and use the boundary conditions to get output. The simulation thus helps in getting results much easily than constructing the real object.

Foresight

In CFD first we make the model then use certain boundary conditions to get the output. Thus using CFD we can give some real world condition say as pressure or temperature and simulate things to get the output. Many variations can be adopted till an optimal result is reached. All of this can be done before physical prototyping and testing.

Efficiency

The analysis gives better idea of, how the object works. So necessary changes could be brought about to facilitate better production of the product. Thus CFD helps to design better and faster bringing about improvement in each step.

The CFD Process

In the past decades many modeling and simulation software have been developed like PHONICS, FLUENT, SRAT-CD, CFX, FLOW -3D and COMPACT. Generally this software's are based on finite volume method. FLUENT is one of the major software which has helped a lot in modeling fluid and heat transfer problems. Complex geometries can be meshed in 2D triangular or quadrilateral, 3D tetrahedral, hexahedral, pyramid, wedge and mixed meshes. Fluent thus help in simulating the unstructured mesh geometries.

FLUENT consist of two parts, one is GMABIT and the other is FLUENT solver. Gambit is used for designing and fluent solver is used for simulation. Gambit helps in meshing the geometries and coarse and fine meshing can be done as per the flow solutions. Once the meshing is done in

Gambit, further modification like setting up boundary condition and executing the solution is done by using Fluent.

There are essentially three stages to every CFD simulation process: preprocessing, solving and postprocessing.

Preprocessing

This is the first step while solving a problem in Fluent. It involves designing a model. The designing is done in Gambit which is the preprocessor of Fluent. Using Gambit, proper geometry whether it is 2D or 3D is produced. Gambit also helps in meshing the model and applying proper boundary conditions. The other major preprocessor except Gambit are TGrid and G/Turbo.

Gambit has many options which make it easier for the person using it. The Boolean Operation helps in converting a solid model to fluid model. Meshing can be done of various types, edge meshing, face meshing and volume meshing. The zones parameter helps in dividing the geometry into different zones as per the problem so that different boundary condition could be set.

Fluent's solvers also couple with other meshing tools such as ANSA, Harpoon, Sculptor and YAMS.

GAMBIT:

On starting the designer care should be taken that you are in the correct directory. Then select the Fluent 5/6 solver. Give a filename to save your project to the above name and then get started with the geometry. First you create a vertex from which you start an edge. Assembling

the edges give the face which in turn gives the volume. Once we obtain the face or volume we can mesh the geometry and can export it to fluent for analysis.

In the upper right portion we have four heads which are geometry, mesh, zones and tool. Click on the geometry which gives the lead to vertex, edge, face, volume and group. Click on the vertex which leads to create vertex which leads to create real vertex button. Enter the coordinates in global form and then click on the Graphics/window control button to get the vertex.

Once all the vertexes are created we can create the edge by joining the vertex. Click on the edge button under geometry. There it asks for the vertex between which the edges have to be drawn. Selecting the edges click on the create edge button to get the edge. Under the geometry button, click on the face button. A floating window called Create Face from Wireframe will appear. Select the edges by holding the shift and clicking the edges from which the face has to be obtained. Click on the create face option and thus we obtain the face.

Once the face is created we need to mesh the file. Under the option command select mesh which give five results. From them select the face option. Click on the top left button in the FACE menu area, the button is called Mesh Faces, and then you will see the mesh in yellow

Then the next work is to set the boundary condition. Under the operation window, click on the zones button. Under the word ZONES two buttons will appear: Specify Boundary Types and Specify Continuum. Click on the Specify Boundary Types button. Make sure that at the top of this window the solver name 'Fluent 5/6' appears. Then select the edge and name it, then select the proper type of condition in which you want to do the simulation.

After the work is over save the file and then export it for further analysis of the design.

Solving

FloWizard is the first software developed by Fluent for the engineers to design and simulate things. POLYFLOW (and FIDAP) are also used in a wide range of fields. The CFD solver does the flow calculations and produces the results. FLUENT is used in most industries. The FLUENT CFD code is very friendly with the user as the design and model conditions can be changed at any moment of time. Thus it efficiently saves time. Fluent software is commonly used because complex models can be made easily. The mesh produced has adaptive and dynamic mesh capability. The graphical user interface (GUI) helps in modeling faster as it helps in better understanding of physics and other functions. Fluent has helped in solving problems with real world conditions. Thus it has excelled in all fields. Some of the fields where it has been found to be useful are:

- multiphase flows
- reacting flows
- rotating equipment
- moving and deforming objects
- turbulence
- radiation
- acoustics, and
- dynamic meshing

The Fluent software has parallel computing ability and hence have greater speed. The software helps in designing the model, simulating it, later helps in postprocessing and getting the final results. Thus it solves larger problems with greater ease.

Postprocessing

Postprocessing is the final step in fluent which allows to get the result in an organized way. It involves interpretation of results in form of images or animations. High resolution images give a better knowledge of the simulation done and hence the output is better interpreted. The postprocessing tool helps to provide desired results.

FLUENT

Fluent in general solve the governing equation of conservation of mass and momentum. It chooses two numerical methods, one is the segregated solver and the other is the couple solver. In the segregated solver the governing equations are solved sequentially while the couple solver solves the governing equation and the transport functions simultaneously.

While solving in fluent, first the mesh file is import and then the grid check is done. Then the solver is selected. Then the energy equation is checked. The material is set and the operating condition is fixed. The boundary condition is set for both inlet and outlet. The monitors are set and the flow field is initialized. Then iteration is done to obtain the converged results.

PROBLEM SOLVING STEPS

After determining the important features of the problem following procedural steps are followed for solving it

1. The geometry model was created and meshing was done.
2. The appropriate solver was started for 2D or 3D modeling.
3. The grid was checked.
4. The solver formulation was selected.
5. The basic equation to solve: laminar or turbulent was selected.
6. The material properties were specified.
7. The boundary properties were specified.
8. The solution control parameter was adjusted.
9. The flow field was initialized.
10. The solution was calculated.
11. The results were examined.
12. The results were saved.
13. If necessary, refine the grid or consider revisions to the numerical or physical model.

Chapter 3

CFD ANALYSIS

CFD MODEL:

A two dimensional axisymmetric model was developed using GAMBIT.

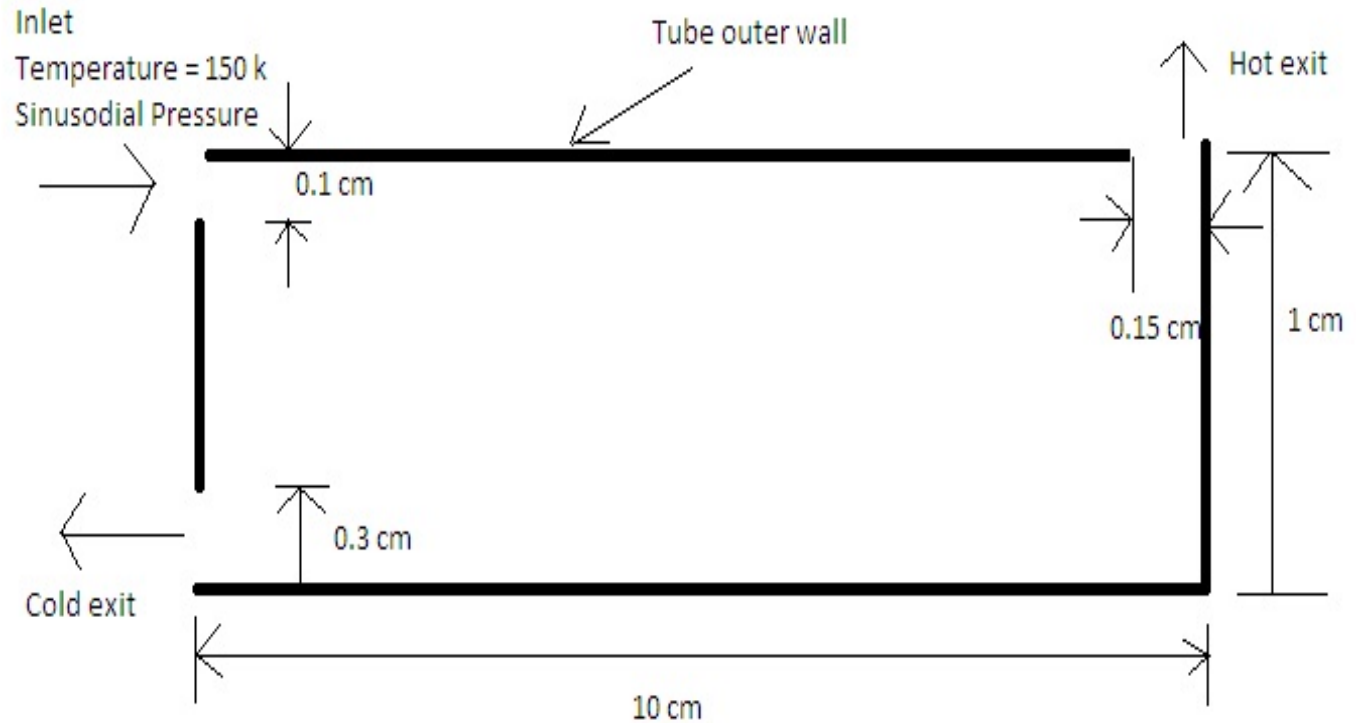


Fig 2 shows the 2D CFD model of vortex tube.

The above model was designed in Gambit with the proper dimensions. The base of the model is the base line or the centre line of the tube. The length of the tube is of 10cm. The inlet is of 1mm diameter. The cold flow outlet is of 3mm. The boundary at the other end holds the hot exit which is of 1.5 mm. The width of the tube is 1cm from the centre line. Air is used as the material to be used in the vortex tube.[3]

PROCEDURE FOR MODELING

STEP 1:

Specify that the mesh to be created is for use with FLUENT 6.0:

Main Menu > Solver > FLUENT 5/6

Verify this has been done by looking in the *Transcript Window* where you should see. The boundary types that you will be able to select in the third step depends on the solver selected.

STEP 2:

Select vertex as tool geometry

TOOL> GEOMETRY> VOLUME

Vertexes at various points are located. The vertex were created at

A= (0, 0),

B= (10, 0),

C= (10, 1),

D= (9.85, 1),

E= (0, 1),

F= (0, 0.3),

G= (0, 0.9)

TOOL> GEOMETRY >EDGE

Then the edges were created using the vertexes. The vertexes were selected using shift and left clicking on the vertexes.

Then edges were created between AB, BC, CD, DE, EG, FG and FA.

TOOL> GEOMETRY >FACE

Then using the edges the face was formed by selecting all the edges together.

STEP 3: MESH

The element is of Tri and the type is pave.

Number of triangular cells= 1309

STEP 4 :(set boundary types)

ZONES > SPECIFY BOUNDARY TYPES.

The edges are selected to assign them particular boundary conditions. The edge EG is the inlet which is of the type mass flow inlet.

The edge FA is the cold flow outlet which is of the type pressure outlet.

The edge CD is the hot flow outlet which is of the type pressure outlet.

STEP 5: (Export the mesh and save the session)

FILE > EXPORT > MESH

File name was entered for the file to be exported. Accept was clicked for 2-D.

Gambit session was saved and exit was clicked.

FILE >EXIT

ANALYSIS IN FLUENT

PROCEDURE:

STEP 1: (GRID)

FILE > READ > CASE

The file channel mesh is selected by clicking on it under files and

Then ok is clicked.

The grid is checked.

GRID > CHECK

The grid was displayed.

DISPLAY > GRID

Grid was copied in ms-word file.

STEP 2 :(Models)

The solver was specified.

DEFINE > MODELS > SOLVER

Solver is segregated

Implicit formulation

Axisymmetric swirl

Time unsteady

DEFINE > MODEL > ENERGY

Energy equation is clicked on.

DEFINE > MODELS > VISCOUS

The standard k- ϵ turbulence model was turned on.

K- ϵ model (2-equation) - Standard

Model constants

C_{mu}-0.09

C₁-Epsilon-1.44

C₂-Epsilon-1.92

Energy Prandtl number= 0.85

Wall Prandtl number= 0.85

KE Prandtl Number 1

No viscosity

STEP 3: (Materials)

By default the material selected was air with properties.

Viscosity, $\mu = 1.7894 \times 10^{-5}$ kg/ms

Density, $\rho = 1.225 \text{ kg/m}^3$.

Thermal Conductivity, $K = 0.0242 \text{ W/mK}$

Specific heat, $C_p = 1.00643 \text{ kJ/kg K}$

Molecular weight = 28.966

STEP 4: (Operating conditions)

Operating pressure = 101.325 KPa

No Gravity.

STEP 5: (Boundary conditions)

DEFINE > BOUNDARY CONDITIONS

INLET

Mass flow rate = 0.006 Kg/s

Total temperature = 150 K

Initial Gauge Pressure is a sinusoidal function.

Axial component of flow direction = 0.8

Radial component of flow direction = 0.31

Tangential component of flow direction = 0.5

Turbulence Kinetic Energy= 1 m²/s²

Turbulence Dissipation Rate= 1m²/s³

HOT OUTLET

Gauge Pressure= 54000 Pa

Backflow temperature= 170 K

Backflow Turbulence Kinetic Energy= 1 m²/s²

Backflow Turbulence Dissipation Rate= 1m²/s³

COLD OUTLET

Gauge Pressure= 150 Pa

Backflow Temperature= 150 K

Backflow Turbulence Kinetic Energy= 1 m²/s²

Backflow Turbulence Dissipation Rate= 1m²/s³

STEP 6: (Solution)

SOLVE > CONTROLS>SOLUTIONS

All flow, turbulent and energy equation used.

Under relaxation factors

Pressure= 0.3

Density= 1

Body Force= 1

Momentum= 0.7

Swirl Velocity= 0.9

Turbulence Kinetic energy= 0.8

Turbulence Dissipation Rate= 0.8

SOLVE >INITIALIZE

Click **INIT**

~Velocity inlet was chosen from the computer from the list.

~ Y and Z component of velocity is 0.

~ Init was clicked and panel was closed.

Three monitors were open for noting the temperature at inlet, hot exit and cold exit.

SOLVE>MONITOR>SURFACE

SOLVE>ITERATE

Time step= 0.005 s

Number of Time step= 5000

Maximum number of iteration per time step=5

Convergence was noticed at 152 iterations

REPORT>SURFACE INTEGRALS

Temperature at inlet, hot exit and cold exit was noted by taking the area weighted average.

The data was saved.

FILE>WRITE>CASE & DATA

STEP 7: (Displaying the preliminary solution)

Display of filled contours of velocity magnitude

DISPLAY>CONTOURS

~ Velocity was selected and then velocity magnitude in the Drop down list was selected.

~filled under option was selected.

~Display was clicked.

Display of filled contours of temperature

DISPLAY>CONTOURS

~Temperature was selected and then Static temperature in the drop down list was selected..

~Filled under option was selected.

~Display was clicked.

Display of filled contours of pressure

DISPLAY>CONTOURS

~ Pressure was selected along various planes

~filled under option was selected.

~Display was clicked.

Display velocity vector.

DISPLAY>VECTOR

~Display was clicked to plot the velocity vectors.

PLOT>XY PLOT

~ Three graphs were plotted.

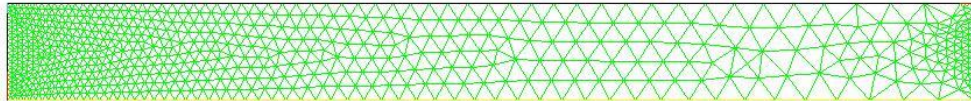
~ One was Static temperature v/s position, another was static pressure v/s position and the third one was velocity magnitude v/s position.

~ The graphs give the magnitude of each at inlet, hot exit and cold exit.

Chapter 4

Result and Discussion

GRID DISPLAY



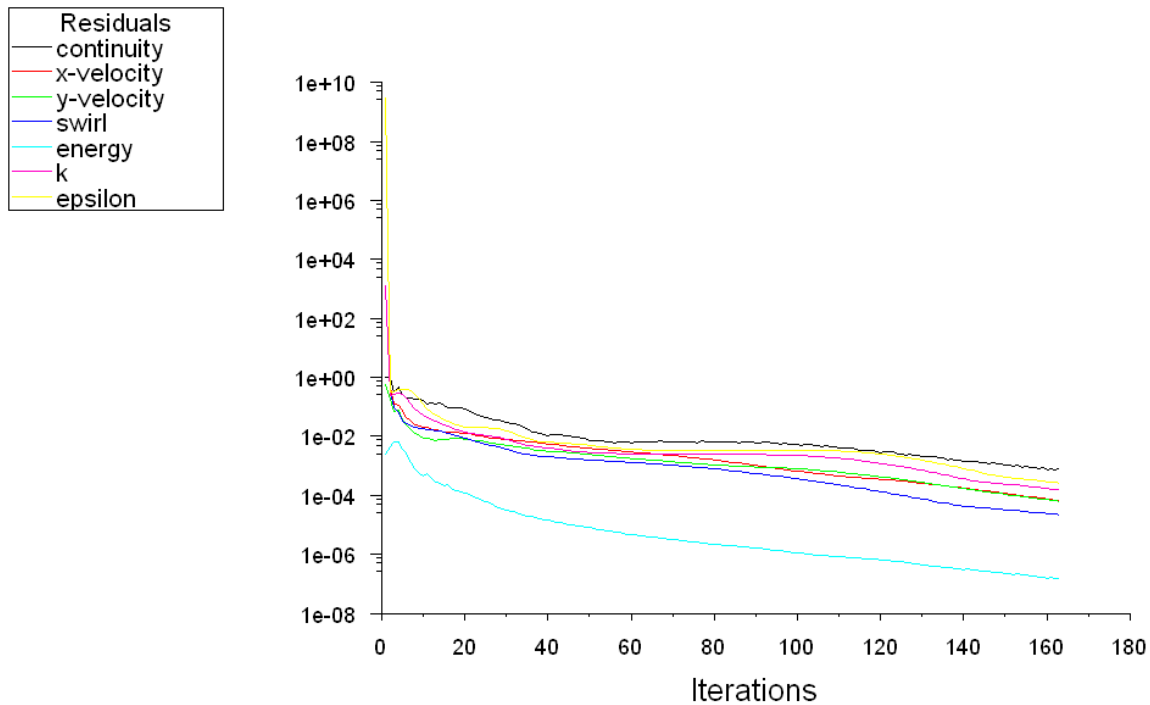
Grid (Time=2.5500e-01)

May 07, 2010
FLUENT 6.2 (axi, swirl, segregated, ske, unsteady)

Fig 4.1 Grid display showing hot outlet, cold outlet and pressure inlet.

The above figure shows the grid pattern of the 2D axisymmetric model. The Grid pattern is created with the Gambit software. The figure shows the mass flow inlet, hot pressure outlet and cold pressure outlet. The lower yellow line is the tube centre line. The fluid flowing in the tube is air. The fluid entered the tube through the inlet. The fluid stream got separated into the hot part which exit through the far end and the cold part which exit through the opening in the near end.

ITERATION CURVE



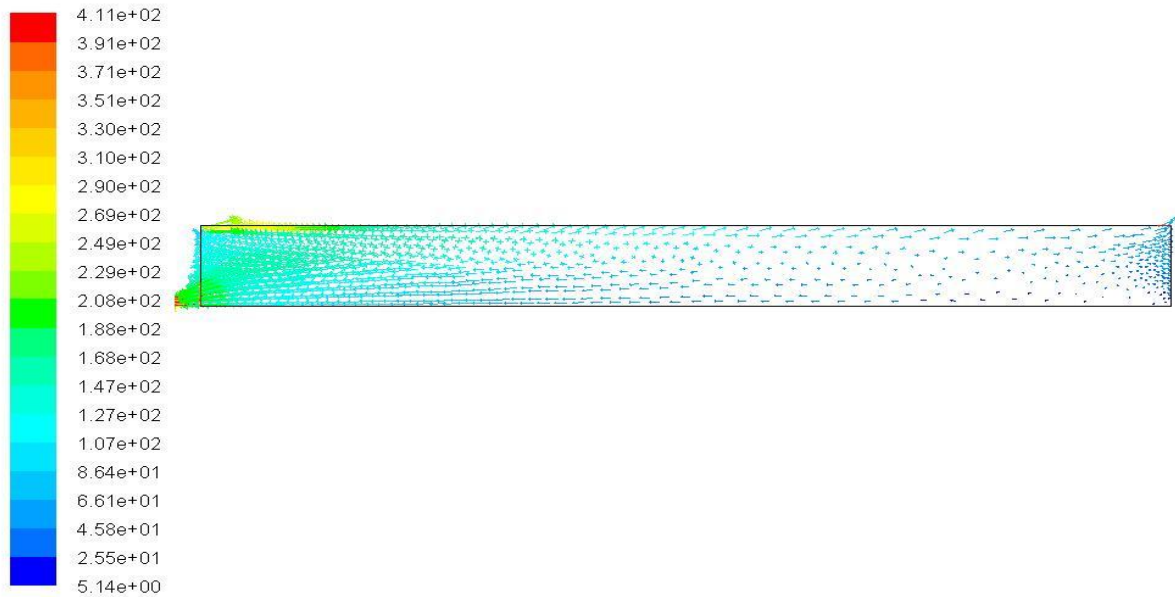
Scaled Residuals (Time=2.1000e-01)

May 05, 2010
FLUENT 6.2 (axi, swirl, segregated, ske, unsteady)

Fig 4.2 Graph shows iteration and convergence.

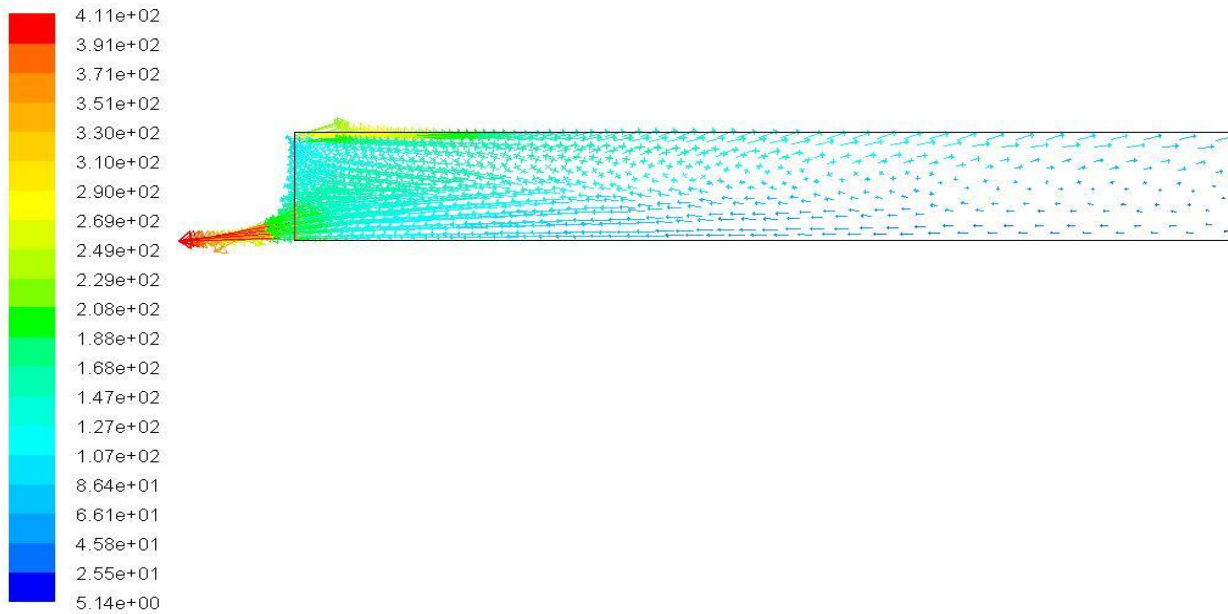
The above figure shows the iteration curve. The solution converges in 152 iterations. The curve shows convergence of various scaled residuals consisting of continuity, x velocity, y velocity, swirl, energy, K and epsilon.

VELOCITY VECTOR



Velocity Vectors Colored By Velocity Magnitude (m/s) (Time=2.1000e-01)

May 05, 2010
FLUENT 6.2 (axi, swirl, segregated, ske, unsteady)

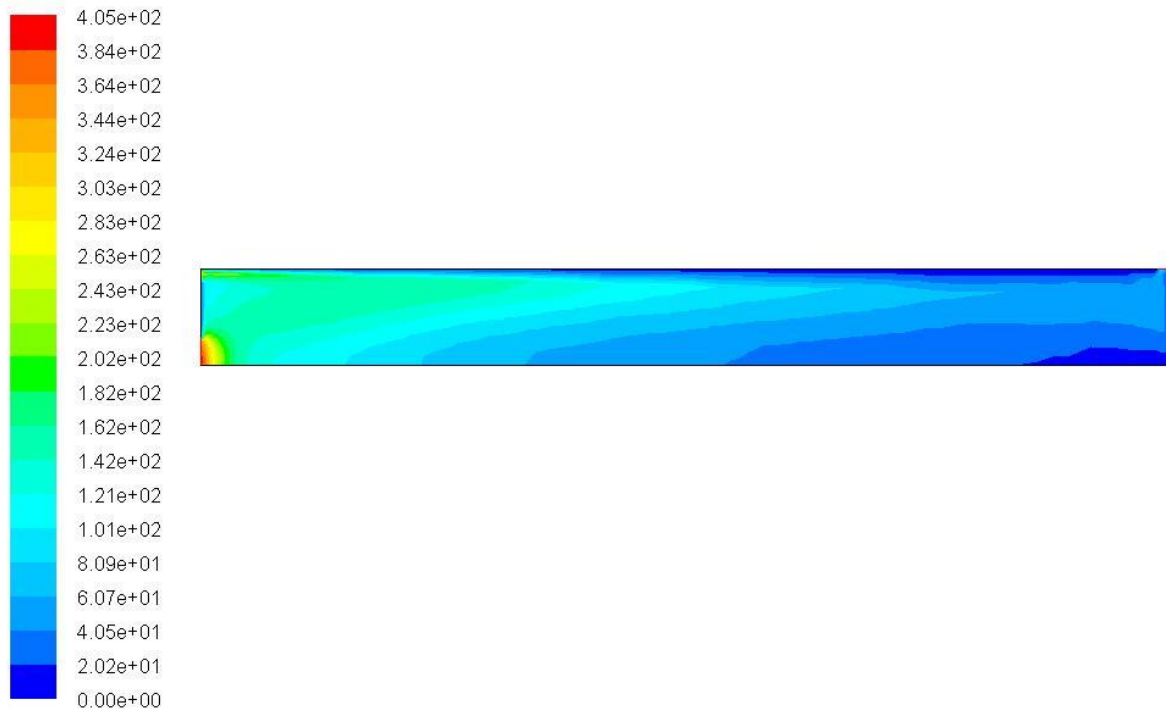


Velocity Vectors Colored By Velocity Magnitude (m/s) (Time=2.1000e-01)

May 05, 2010
FLUENT 6.2 (axi, swirl, segregated, ske, unsteady)

Fig 4.3(a) and Fig 4.3(b) shows Velocity vector.

VELOCITY CONTOUR



Contours of Velocity Magnitude (m/s) (Time=2.1000e-01)

May 05, 2010
FLUENT 6.2 (axi, swirl, segregated, ske, unsteady)

Fig 4.4 Contour showing the velocity variation along inlet, cold outlet and hot outlet.

TEMPERATURE CONTOUR



Contours of Static Temperature (k) (Time=1.8500e-01)

May 01, 2010
FLUENT 6.2 (axi, swirl, segregated, ske, unsteady)

Fig 4.5 Contour showing the temperature variation along inlet, cold outlet and hot outlet.

GRAPHS

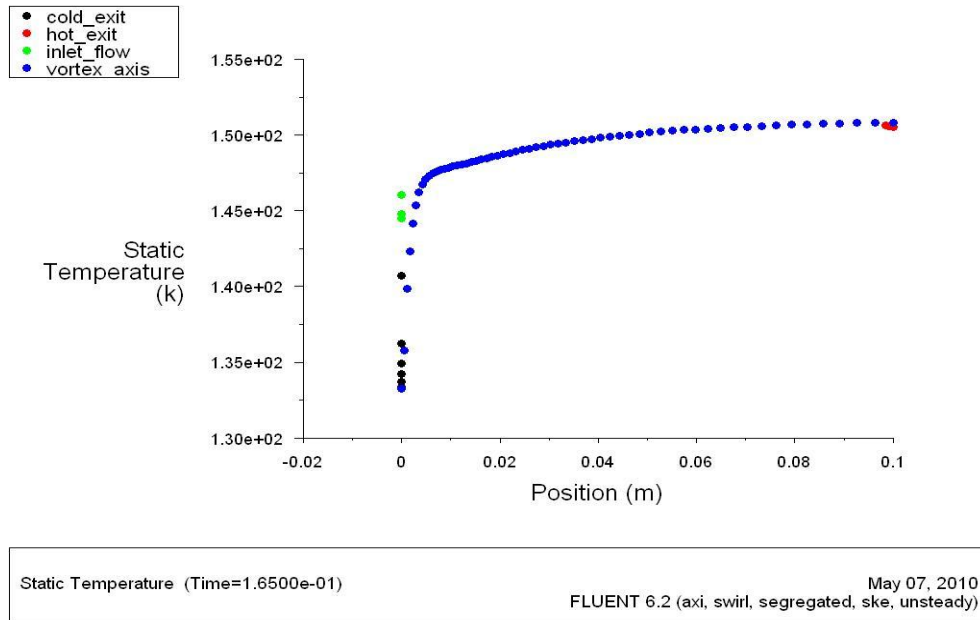


Fig 4.6: Graph showing the Temperature distribution along the vortex axis, inlet and both the outlets.

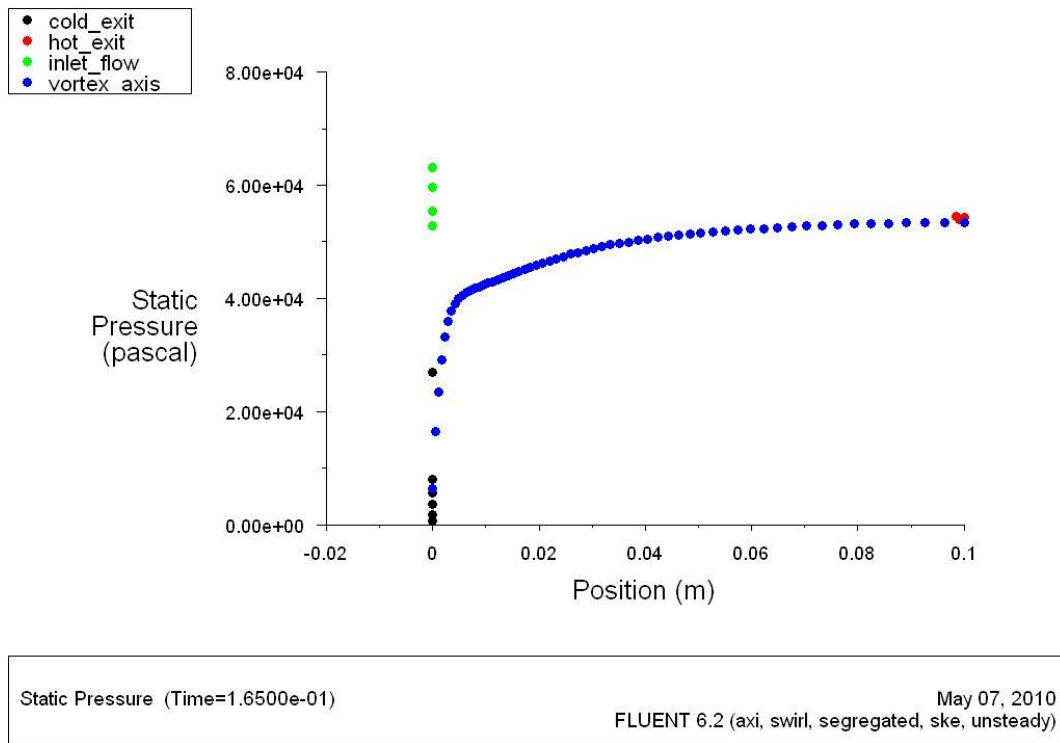


Fig 4.7: Graph showing the Pressure distribution along the vortex axis, inlet and both the outlets.

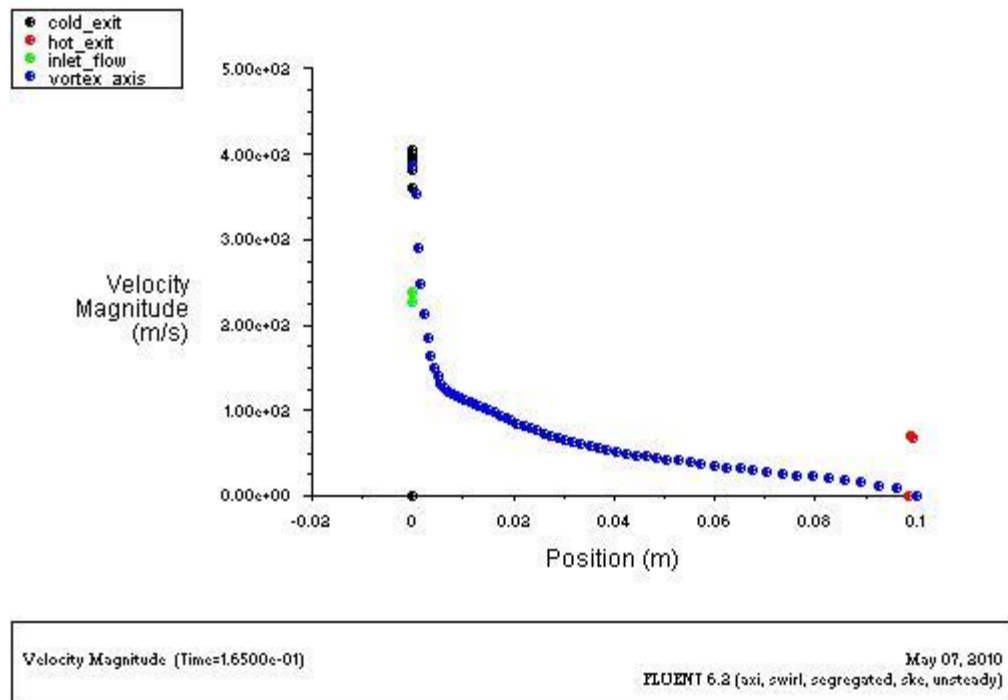


Fig 4.8: Graph showing the Velocity distribution along the vortex axis, inlet and both the outlets.

DISCUSSION

VELOCITY VECTOR AND CONTOUR

FIGURE 4.3(a) and 4.3(b)

The figure shows velocity vector variation along inlet, hot and cold outlet. The fluid enters the vortex tube through the inlet with swirl motion and a very low radial velocity component. The fluid loses velocity as it moves in the axial direction and reaches a point where the velocity becomes zero. This point is known as stagnation point. From the stagnation point, reverse flow is observed and the fluid moves out through the cold exit. The fluid which moves forward in the axial direction moves out through the hot exit.

FIGURE 4.4

The figure shows the contour of velocity along inlet, hot and cold outlet. The fluid entering the vortex tube has a very low radial component of velocity. The contour of velocity clearly indicates the stagnation point. From the contour it can be inferred that velocity at hot outlet is very low compared to that at cold outlet.

CONTOURS OF TEMPERATURE

FIGURE 4.5

The figure shows contours of temperature variation along inlet, hot and cold outlet. Due to cooling effect of inlet air from inlet, the temperature of the fluid increases as we move from left to the right. This can be represented by yellow color which changes to red as we move left. The

fluid moving out through the cold outlet has a low temperature compared to both inlet and hot outlet and is marked with a blue color in the contour display.

GRAPHS

FIGURE 4.6

The graph in figure shows the temperature distribution along the vortex axis, inlet and both the outlets. As we see, the maximum temperature is observed at the hot outlet. The fluid moving out through the cold outlet has the lowest temperature in the vortex tube. Hence we observed a refrigeration effect when a fluid is tangentially introduced into the vortex tube.

FIGURE 4.7

The graph in figure shows the pressure distribution along the vortex axis, inlet and both the outlets. A sinusoidal pressure function was provided at the inlet. The pressure at both the cold and hot outlet was fixed. The pressure at the hot outlet was more than that of the cold outlet.

FIGURE 4.8

The graph in figure shows the velocity distribution along the vortex axis, inlet and both the outlets. The velocity is almost zero at hot outlet. As the fluid moves from inlet to hot outlet in the axial direction it loses its speed and drops to zero. Reverse flow occurs and hence the velocity increases as it moves towards the cold outlet. Thus the velocity at cold outlet is high.

Chapter 5

Conclusion

CONCLUSION

The solution was observed after 152 iterations when it converges. The velocity vector and the velocity contour were displayed. The temperature distribution was observed. The final temperature at the inlet, hot outlet and the cold outlet were noted when the area weighted average of static temperature was taken.

These are the final report of temperature being calculated

Cold Exit= 135.81826 K

Hot Exit= 150.58805 K

Inlet flow= 144.80929 K

Net= 146.42968 K

Thus, from simulating the data we observe that a refrigeration effect is produced when a fluid is tangentially introduced into the vortex tube. The fluid in the tube separates into two parts; one is the hotter part which flows out through the hot exit and the cold part which flows out through the cold exit. It is also observed that as the fluid moves axially the velocity decreases and reaches a stagnation point from which reverse flow occurs.

Chapter 6

References

REFERENCES

1. Behera Upendra, Paul P.J, Dinesh K, Jacob S., Numerical investigation on flow behavior and energy separation in Ranque-Hilsch vortex tube.
2. Eiamsa-ard Smith, Promvonge Pongjet, Numerical investigation of the thermal separation in a Ranque-Hilsch vortex tube.
3. Aljuwayhel N.F, Nellis G.F, Klein S.A, Parametric and internal study of the vortex tube using a CFD model.
4. Eiamsa-ard Smith, Promvonge Pongjet, Review of Ranque–Hilsch effects in vortex tubes.
5. Eiamsa-ard Smith, Promvonge Pongjet, Numerical simulation of flow field and temperature separation in a vortex tube.
6. Frohlingsdorf W, Unger H, Numerical investigation of the compressible flow and the energy separation in the Rangué-Hilsch vortex tube.
7. Dincer K, Baskaya S, Uysal B.Z, Experimental investigation of the effect of length to diameter ratio and nozzle number on the performance on counter flow Rangué-Hilsch vortex tube.
8. Behera Upendra, Paul P.J, Dinesh K, Jacob S, Kasthuriangan, Ram S.N, CFD analysis and experimental investigations towards optimizing the parameters of Ranque-Hilsch vortex tube.
9. Uluer Onuralp, Kirmaci Volkan, An Experimental investigation of the cold mass fraction, nozzle number, and inlet pressure effects on performance of counter flow vortex tube.

10. Farouk Tanvir, Farouk Bakhtier, Gutsol Alexander, Simulation of gas species and temperature separation in the counter-flow Ranque-Hilsch vortex tube using the large eddy simulation technique.
11. Srivastava Shivam, Saraogi Mayur, Electronic chip cooling in horizontal configuration using Fluent-Gambit.
12. Wu Y.T, Ding Y, Ji Y.B, Ma C.F, Ge M.C, Modification and experimental research on vortex tube.
13. Date Anil W., Introduction to computational fluid dynamics.